FINITE ELEMENTS SAVE CYLINDERS

This case study provides an excellent example of a situation where, thru the use of computer simulation, many of the hours spent in experimentation and prototype construction of various modification or experimental designs were avoided. A solution was obtained to a problem of fatigue fractures.

 Written by Anthony Metoyer, a student at the Illinois Institute of Technology, for the American Society of Mechanical Engineers Case Contest 1977. The cooperation of Clark Equipment Company is gratefully acknowledged.

FINITE ELEMENT ANALYSIS OF HYDRAULIC CYLINDER BASE WELDS

During the spring of 1975, several of the hydraulic cylinders on the tractor loaders were returned to the lab from the field, because of fractures in the base cap / tube wall weld. Examination of these cylincers revealed that the problem stemed from a fatigue crack that began at the root of the weld and propagated to the outside surface. (see fig. 1)

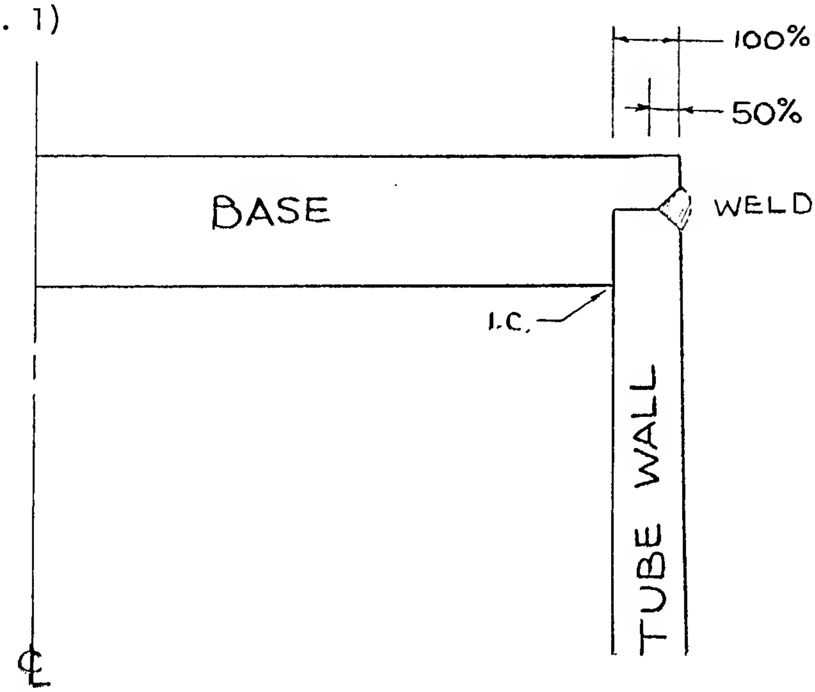


Figure 1

The cause of these failures was initially attributed to the 50-60% weld penetration of the tube wall. It was felt that this shallow weld penetration should be increased to possibly a 100% penetration or even a full penetration to the inside corner, to reduce the stresses in this area.

However, previous experimental work revealed a strong possibility that a modification of the penetration alone, would not provide the required reduction in the stresses in this area. Tests revealed that only a marginal improvement in the number of life cycles could be obtained. Regardless of this data, questions were raised as to the effectiveness of an increased weld penetration, as a solution to this problem.

In light of these questions, it was decided to perform a finite element analysis on the problem. The finite element method uses a computer generated mathmatical model of an experimental design or modified component, and simulates the required experiments or tests mathmatically. The resulting reactions, displacements and stress concentrations are then computed and available as information for further analysis of the design or prototype construction.

Using the finite element method; 50%, 100% and full penetration to the inside corner weld models were analized using a 125% load factor. (see figures 2,3, &4 respectively) The stress concentrations are shown in the areas of a high density in the number of isostress lines.

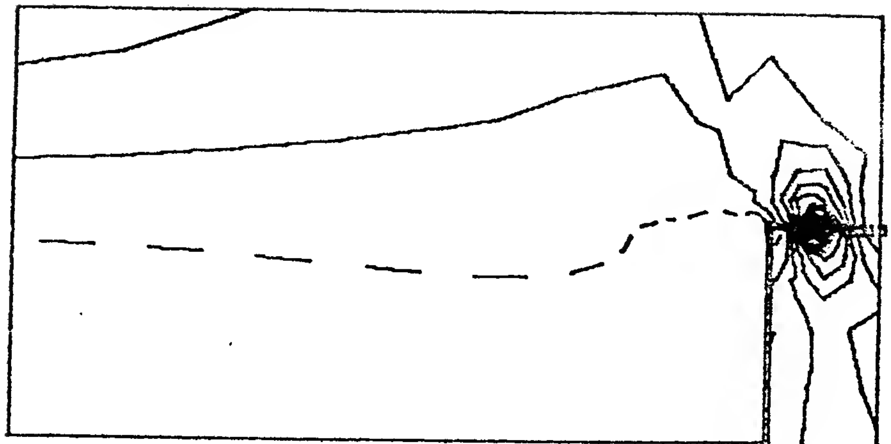
These modifications revealed only a 10% change over the original design. A much higher improvement was needed, not only for present use, but for future designs using increased pressures. Reexaminization of the models revealed the problem source to be the high stresses created due to bending at the weld region. Regardless of where the weld was placed, maximum stress concentrations occured in this region.

To reduce this effect it was decided to move the maximim bending away from the worst notch. This was accomplished by providing a region in the base cap with a smooth radius on the inside surface. (see fig. 5)

The analysis of this design (fig. 6) was based on a 250% load factor. A 75% reduction in stresses was obtained as well as the reduction of stress concentration in the worst notch area. Another feature of this new design is its ability to be easily adapted to current materials and processes.

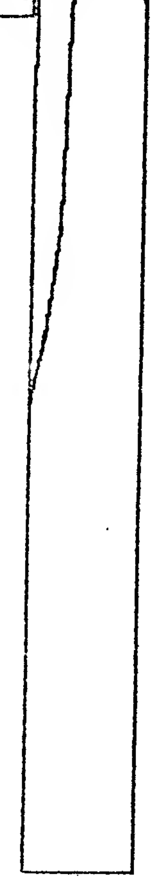
In conclusion, it can be seen that with the use of computer simulation, a fairly accurate and quick solution can be obtained to a given problem. The construction of a prototype to evaluate each of the possible designs is not only costly, but time consuming when a solution is required for production to continue.

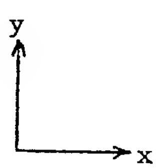
Isostress Lines
= 4000 psi



Pressure = 2750 psi
50% Penetration
Current Design
Max Stress = 62 ksi
Min Stress = -4 ksi

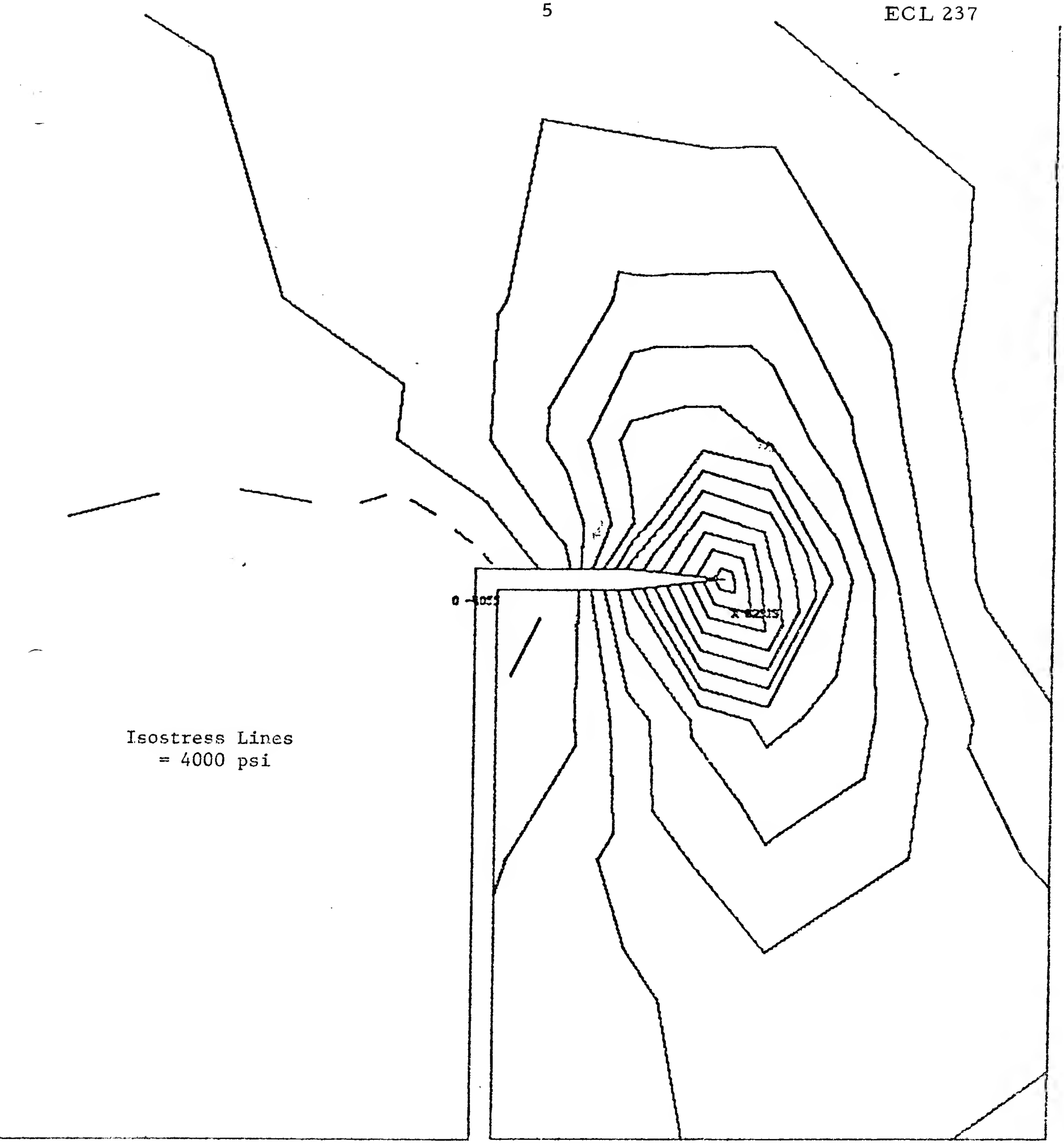
The primary component of the maximum stress is the y-stress, which is perpendicular to the cracks observed in failure analyses.





Maximum Stress

Figure 2-A



Blown up view of weld root

Figure 2-B

MAXIMUM STRESS

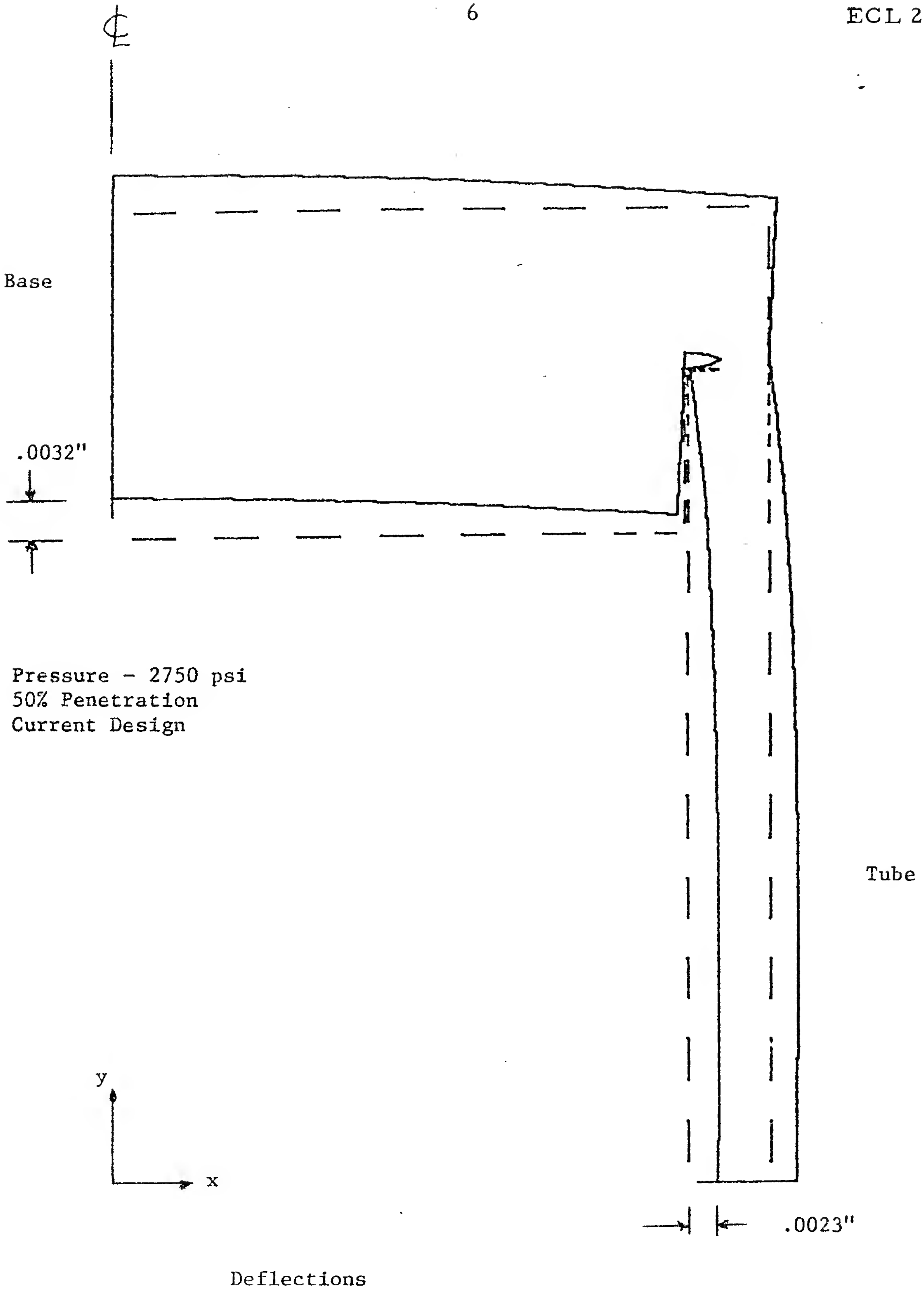
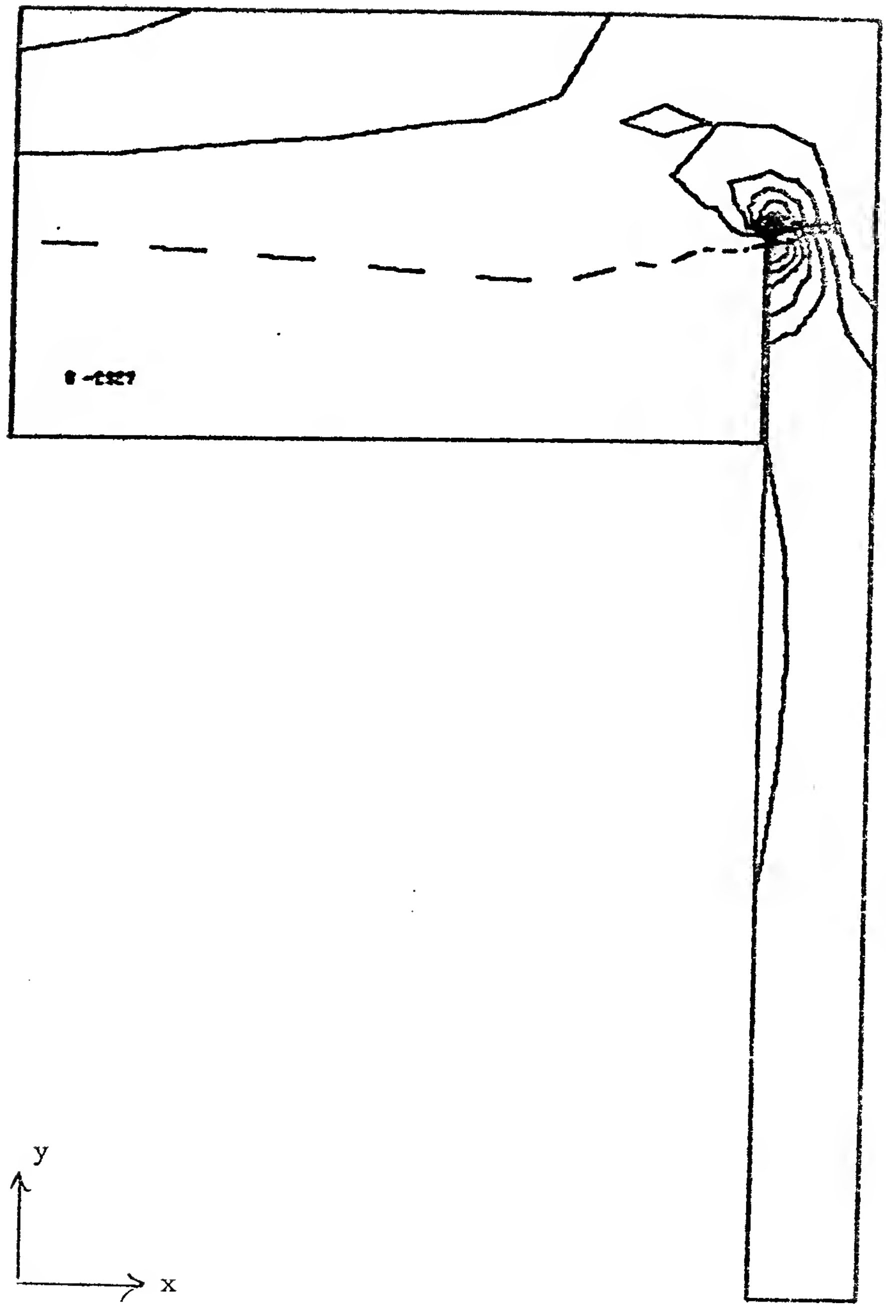

℄
Base
.0032"
Pressure - 2750 psi
50% Penetration
Current Design
y
x
Tube
.0023"
Deflections

Figure 2-C

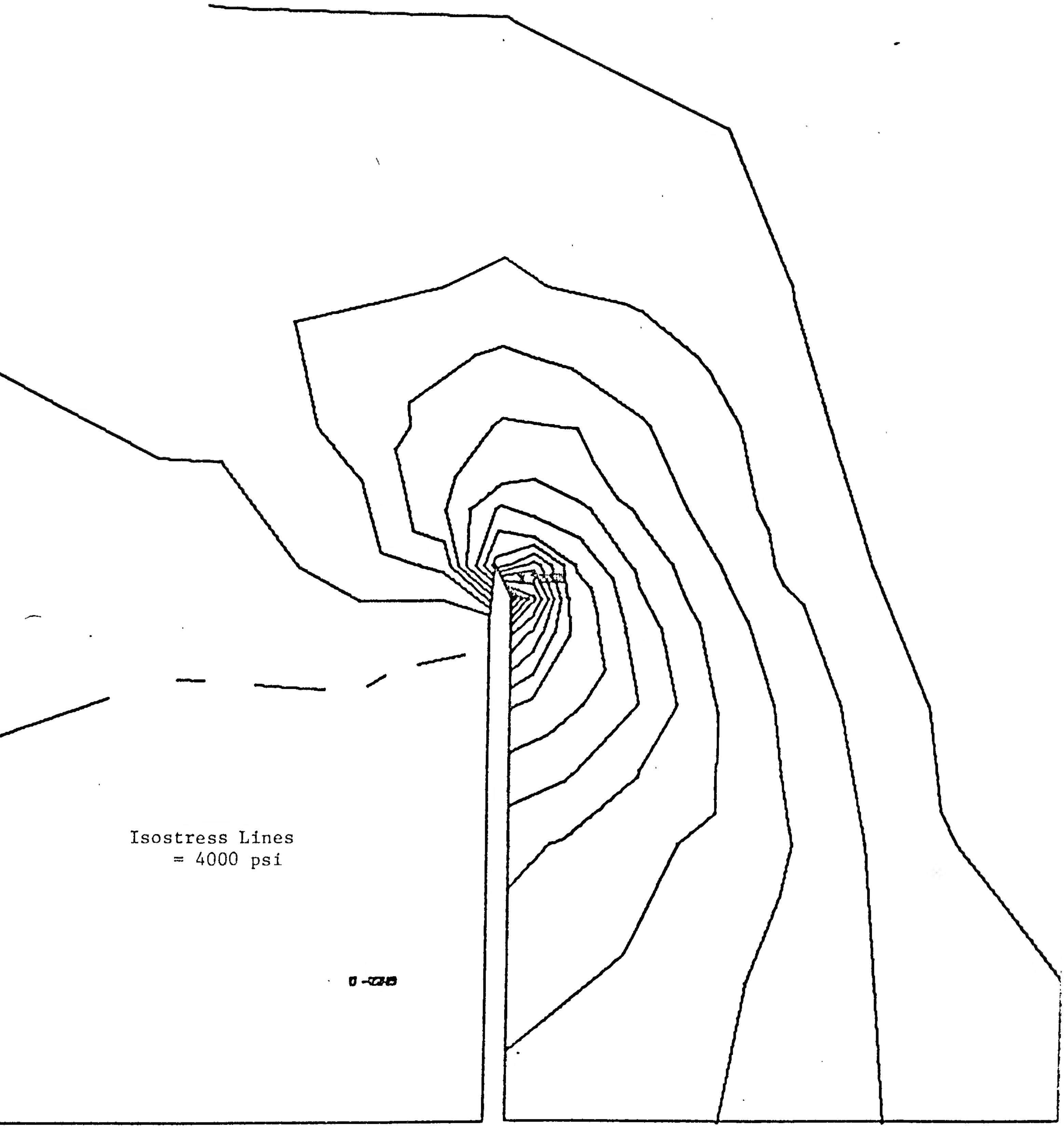
Isostress Lines
= 4000 psi



Pressure = 2750 psi
Full Penetration
Max Stress = 72 ksi
Min Stress = -3 ksi

Maximum Stress

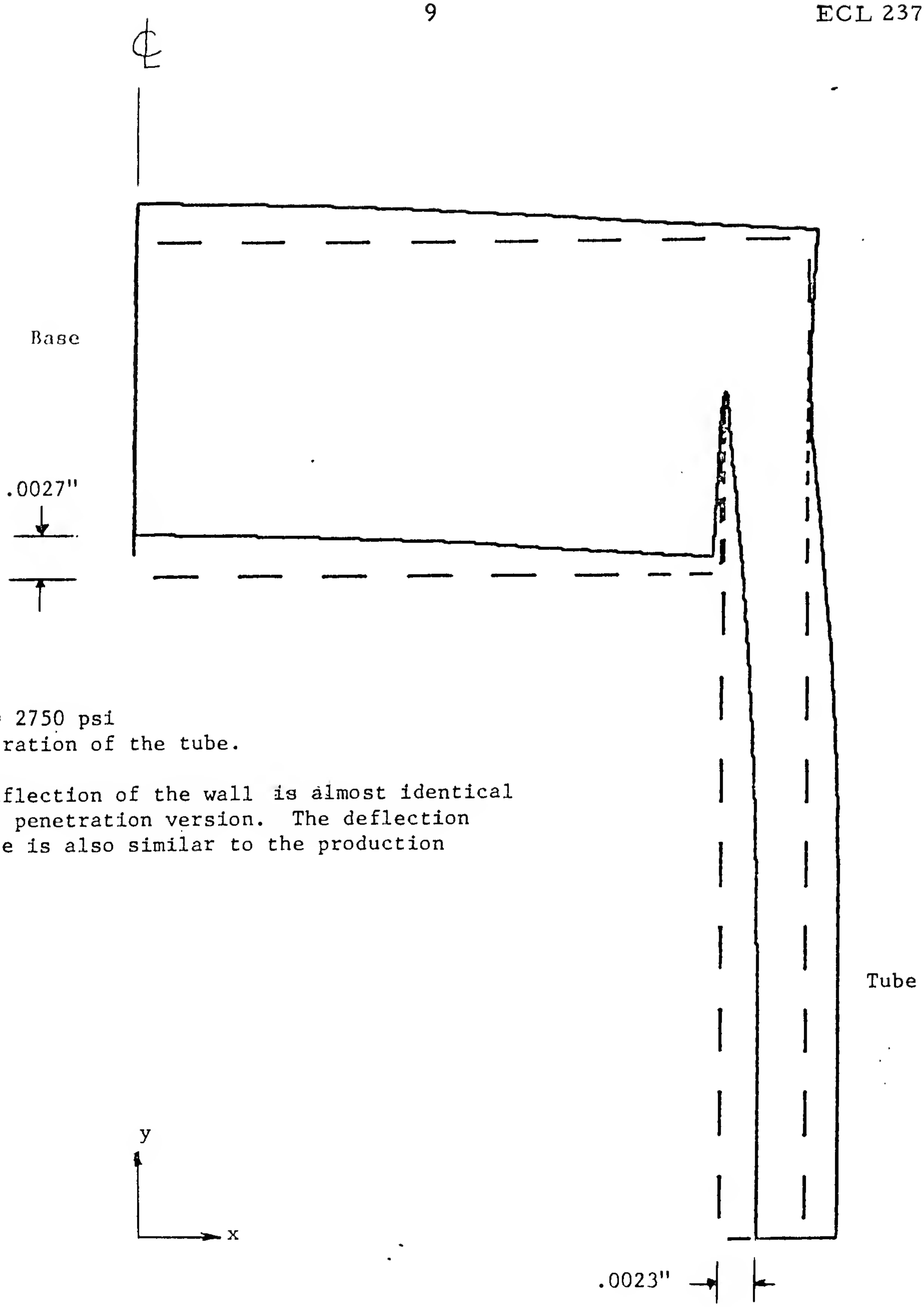
Figure 3-A



Blown up view of weld root

Figure 3-B

MAXIMUM STRESS



Pressure = 2750 psi
Full penetration of the tube.

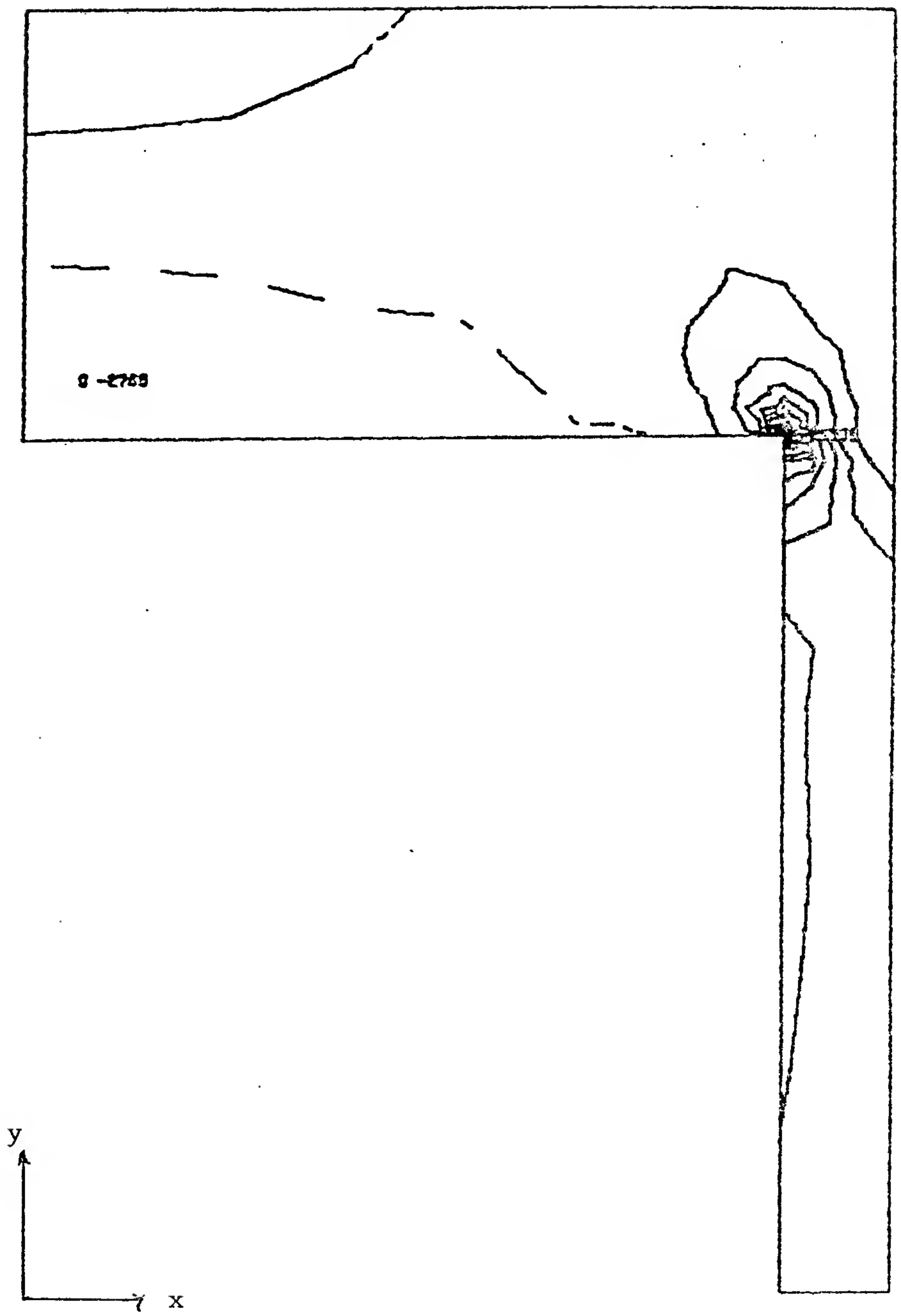
The deflection of the wall is almost identical to the 50% penetration version. The deflection of the base is also similar to the production version.

Deflections

Figure 3-C

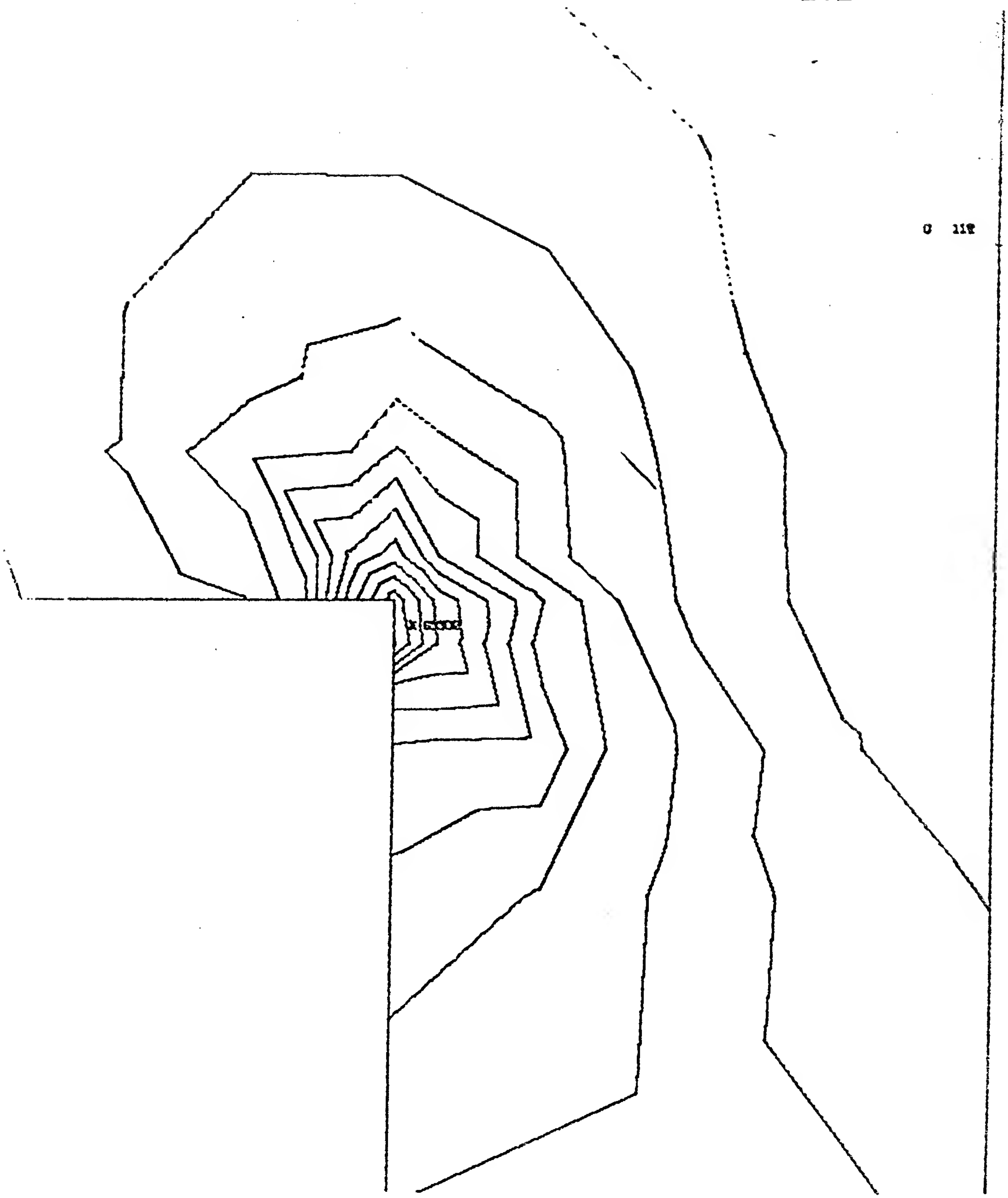
Isostress Lines
= 4000 psi

Pressure = 2750 psi
I.D. Penetration
Max Stress = 63 ksi
Min Stress = -3 ksi



Maximum Stress

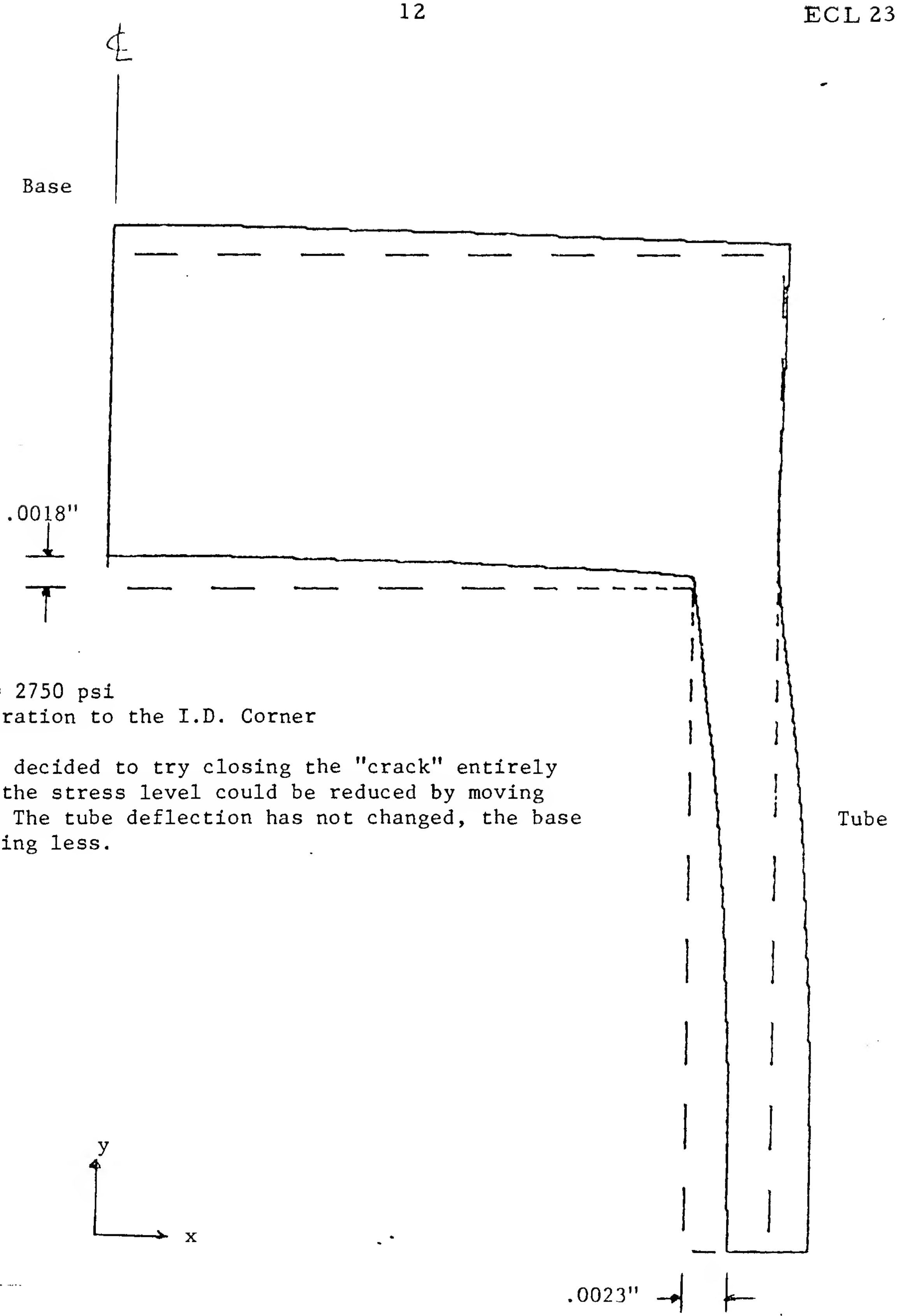
Figure 4-A



Blown up View of Corner

Figure 4-B

MAXIKUM SIRESS

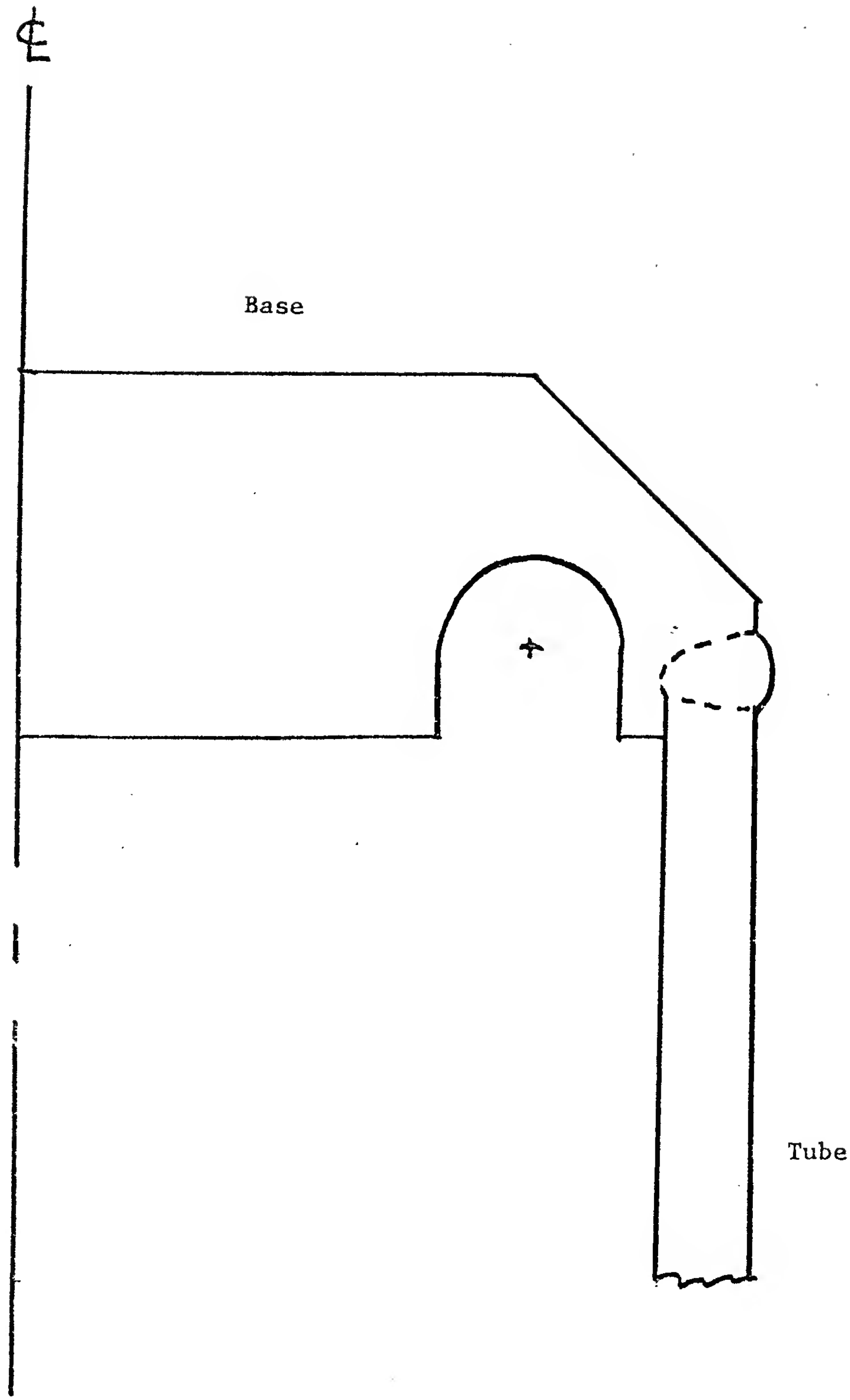


Pressure = 2750 psi
Full Penetration to the I.D. Corner

It was decided to try closing the "crack" entirely to see if the stress level could be reduced by moving the weld. The tube deflection has not changed, the base is deflecting less.

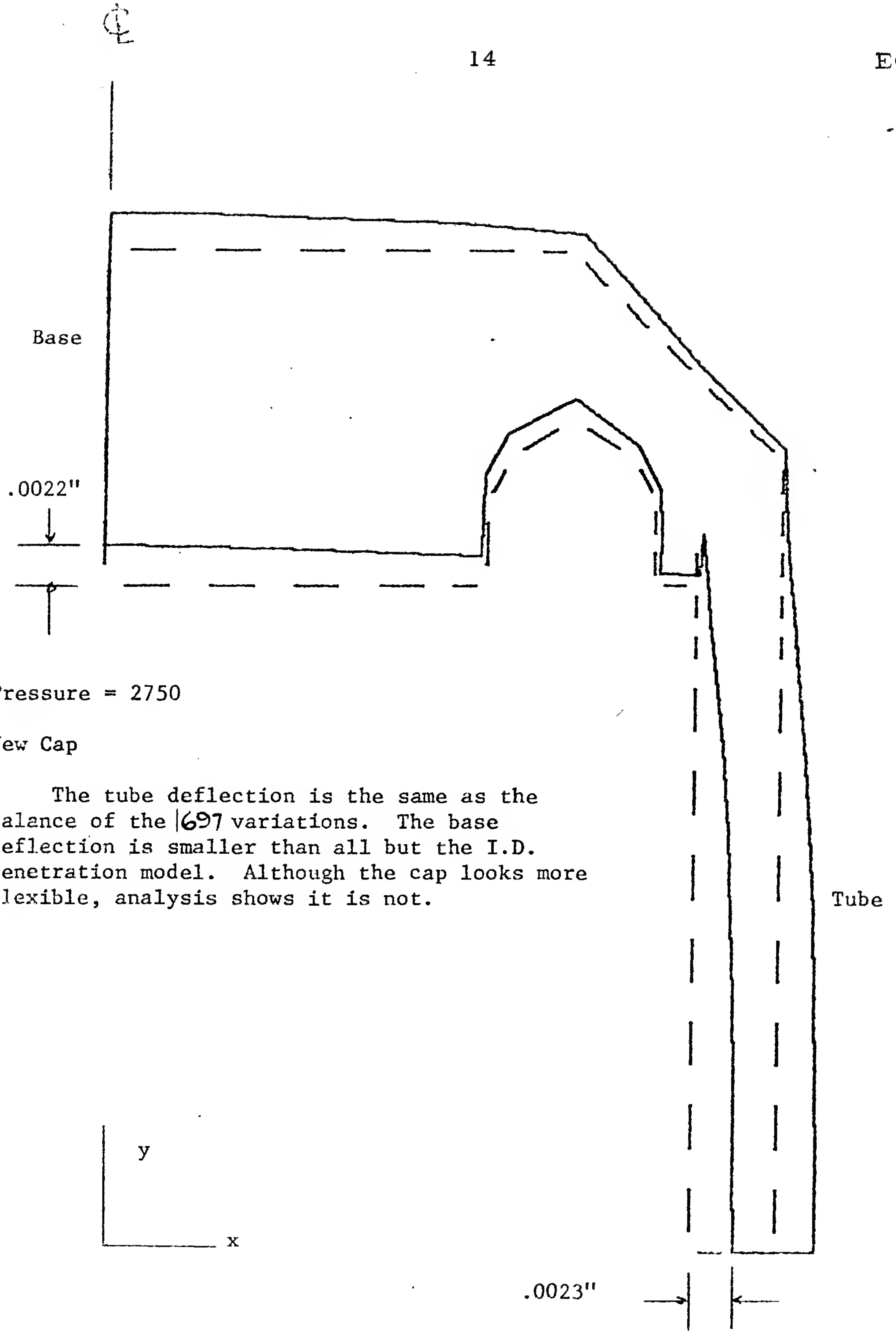
Deflections

Figure 4-C



New Cap Design

Figure 5



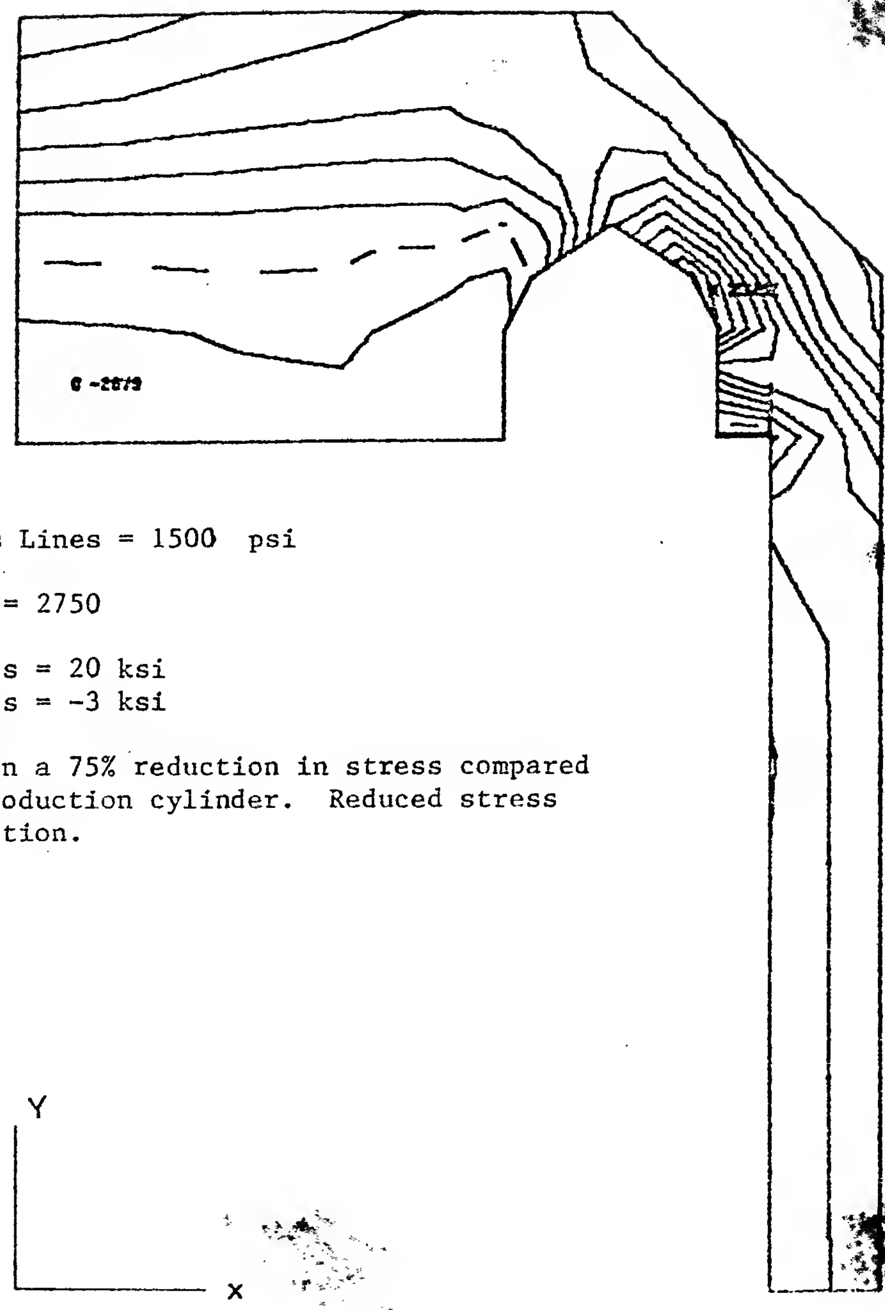
Pressure = 2750

New Cap

The tube deflection is the same as the balance of the 697 variations. The base deflection is smaller than all but the I.D. penetration model. Although the cap looks more flexible, analysis shows it is not.

Deflections

Figure 6-A



Isostress Lines = 1500 psi

Pressure = 2750
New Cap
Max Stress = 20 ksi
Min Stress = -3 ksi

Again a 75% reduction in stress compared to the production cylinder. Reduced stress concentration.

Maximum Stress

Figure 6-B